Faculty of Engineering

EXPERIMENTAL STUDY OF FLUID FLOW IN A DOUBLE L-BEND PIPE WITH SMALL NOZZLE OUTLETS.

Norikhsan Bin Haji Anuar

Bachelor of Engineering with Honours
(Mechanical Engineering and Manufacturing Systems)
2006

EXPERIMENTAL STUDY OF FLUID FLOW IN A DOUBLE L-BEND PIPE WITH SMALL NOZZLE OUTLETS.

NORIKHSAN BIN HAJI ANUAR

This project is submitted in partial fulfillment of the requirements for the degree of
Bachelor of Engineering with Honours
(Mechanical and Manufacturing System)

Faculty of Engineering
UNIVERSITY MALAYSIA SARAWAK
2006

UNIVERSITI MALAYSIA SARAWAK

fk

BORANG PENGESAHAN STATUS TESIS

Judul: **EXPERIMENTAL STUDY OF FLUID FLOW IN A DOUBLE L-BEND PIPE WITH SMALL NOZZLE OUTLETS**

SESI PENGAJIAN: 2005/2006

Saya **NORIKHSAN BIN HAJI ANUAR**
(HURUF BESAR)

mengaku membenarkan tesis * ini disimpan di Pusat Khidmat Maklumat Akademik, Universiti Malaysia Sarawak dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hakmilik Universiti Malaysia Sarawak.
2. Pusat Khidmat Maklumat Akademik, Universiti Malaysia Sarawak dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Membuat pendigitan untuk membangunkan Pangkalan Data Kandungan Tempatan.
4. Pusat Khidmat Maklumat Akademik, Universiti Malaysia Sarawak dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
5. ** Sila tandakan (√) di kotak yang berkenaan

☐ SULIT (Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972).

☐ TERHAD (Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/ badan di mana penyelidikan dijalankan).

☐ TIDAK TERHAD

Disahkan oleh

(TANDATANGAN PENULIS)

(TANDATANGAN PENYELIA)

Alamat tetap: Lot 1068, Jalan Piasau Jaya 2B,
Piasau Jaya Fasa 2,
98000, Miri, Sarawak.

IR. DR. ANDREW R.H. RIGIT
(Nama Penyelia)

Tarikh: 29/05/06.

Tarikh: 5.6.6

CATATAN * Tesis dimaksudkan sebagai tesis bagi Ijazah Doktor Falsafah, Sarjana dan Sarjana Muda.
** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/ organisasi berkenaan dengan menyatakan sekali sebab dan tempoh tesis ini perlu dikelaskan sebagai SULIT dan TERHAD.

APPROVAL SHEET

This Final Year Project report entitled **"Experimental study of fluid flow in a double L-bend pipe with small nozzle outlets"** prepared and submitted by **Norikhsan bin Haji Anuar** as a partial fulfillment of the requirement for the degree in Bachelor of Engineering with Honours in Mechanical Engineering and Manufacturing Systems is hereby read and approved by:

5.6.6

Ir. Dr. Andrew R.H. Rigit

Project Supervisor

Date

Dedicated to my beloved family and Neera.

ACKNOWLEDGEMENT

First and foremost, *Alhamdulillah*, May Allah peace and blessing upon Prophet Muhammad, his family and companions. I would like to take this opportunity to thank number of people that have contributed direct or indirectly in completing my final year project successfully.

I would like to express my first gratitude and great appreciation to my supervisor; Ir. Dr. Andrew R. H. Rigit for his excellent guidance, supervision and full support throughout the completion of this project.

Thanks also to the lecturers and staff of the Faculty of Engineering of University Malaysia Sarawak for their guidance, help and advice in completing this project and for creating a stimulating atmosphere.

Finally, I would like to thanks to my beloved family mostly, and my friends for their support, encouragement and help from the beginning till the end of this project.

ABSTRACT

Fluid flow in circular pipes is commonly encountered in practice. Therefore, a proper understanding of the mechanic of fluids is extremely important in many areas of engineering nowadays. This report gives an introduction on fluid flow characteristics and problem faced due to non-uniform fluid discharge from the pipe nozzle outlets. The objective of this study is to validate fluid flow simulation using a previous computational fluid dynamics result and to estimate the fluid discharge from the present design of pipe nozzle outlets. This report gives some information on how to find and calculate the flow rate, pressure and velocity of the fluid in and discharge from the pipe. Some equation that is usually been used will be introduce later in this report. The method and the experiment setup are included and have been carried out accordingly. The result obtained from the experiment showed that the computational fluid dynamic result is valid. Some modification on the pipe is needed to get uniform fluid discharge from the pipe nozzle outlets.

ABSTRAK

Aliran bendalir dalam paip sperikal secara umumnya selaras dengan amalan. Oleh yang demikian, pemahaman yang teliti tentang mekanik bendalir adalah amat penting dalam kebanyakan bidang kejuruteraan pada masa kini. Laporan ini menunjukkan pengenalan cirri-ciri aliran bendalir dan masalah yang dihadapi disebabkan ketidakseragaman bendalir yang keluar dari nozel paip. Objektif kajian ini adalah untuk mengesahkan keputusan yang diperolehi dari simulasi aliran bendalir yang sedia ada menggunakan pengkomputeran mekanik bendalir dan untuk menganggarkan pengeluaran bendalir dari rekabentuk nozel paip yang sedia ada. Laporan ini memberi beberapa maklumat untuk mencari dan mengira kadar aliran, tekanan dan halaju bendalir di dalam dan yang keluar dari paip. Beberapa persamaan yang biasa digunakan akan diperkenalkan kemudian dalam laporan ini. Kaedah dan susunan eksperimen ada disertakan dan telah dijalankan sewajarnya. Keputusan yang diperolehi dari eksperimen ini menunjukkan yang keputusan pengkomputeran aliran bendalir adalah sah. Beberapa modifikasi ke atas paip adalah perlu untuk mendapatkan keseragaman bendalir yang keluar dari nozel paip.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

FIGURE	TITLE	PAGE

LIST OF NOMENCLATURE

Q	-	Flow rate (litre/min)
V	-	Velocity (m/s)
A	-	Area ($\frac{1}{4}\pi d^2$)
P	-	Pressure (kg/m^2)
ρ	-	Density (kg/m^3)
g	-	Gravity (m/s^2)
D	-	Diameter (m)
μ	-	Absolute viscosity (kg/m.s)
ν	-	Kinematic of viscosity (m^2/s)
Re	-	Reynolds number
V	-	Volume (m^3)
h/x	-	Height (m)
t	-	Time (min) or (s)
F	-	Force (N) or (kg.m/s^2)
m	-	Mass (kg)
θ	-	Angle

CHAPTER 1

INTRODUCTION

1.1 Introduction

Fluid flow can be classified as external or internal flow, depending on whether the fluid is forced to flow over a surface or in a conduit. According to Potter and Wiggert (1997), fluids are those liquids and gases that move under the action of a shear stress, no matter how small that shear stress may be. This means that even a very small shear stress results in motion in the fluid. Typically, liquids are considered to be incompressible, whereas gases are considered to be compressible. However, there are exceptions in everyday engineering applications.

Fluid flow in circular and noncircular pipes is commonly encountered in practice. The hot and cold water that used in houses is pumped through pipes. Water in a city is distributed by extensive piping networks. Oil and natural gas are transported hundreds of miles by large pipelines. The cooling water in an engine is transported by hoses to the pipes in the radiation where it is cooled as it flows. Thermal energy in a hydronic space heating system is transferred to the circulating water in the boiler and then it is transported to the desired locations through pipes.

A proper understanding of the mechanics of fluids is extremely important in many areas of engineering nowadays. In biomechanics the flow of blood and cerebral fluid are of particular interest, in meteorology and ocean engineering an understanding of the motions of air movements and ocean currents requires a knowledge of the mechanics of fluids, chemical engineers must understand fluid mechanics to design the many different kinds of chemical-processing equipment, mechanical engineers design pumps, turbines, internal combustion engines, air compressors and air-conditioning equipment using a proper understanding of fluids mechanics.

Fluid flow through pipes or ducts is commonly used in heating and cooling applications and fluid distribution networks. The fluid in such application is usually forced to flow by a fan or pump through a flow section. Although the theory of fluid flow is reasonably well understood, theoretical solutions are obtained only for a few simple cases such as fully developed laminar flow in a circular pipe. Therefore, we must rely on experimental results and empirical relations for most fluid flow problems rather than closed-form analytical solutions.

Nevertheless, it is quite important to know the characteristic and the properties of the fluid because the fluid can be either laminar or turbulent flow, compressible or incompressible, and also can be dependent on the variations of length and diameter of the pipe. The variables that are also needed to define a fluid and its environment are such as the pressure, temperature, velocity, density, viscosity and also time.

1.2 Historical Note

The science of fluid mechanics began with the need to control water for irrigation purposes in ancient Egypt, Mesopotamia and India. Although these civilizations understood the nature of channel flow, there is no evidence that any quantitative relationship had been developed to guide them in their work. It was not until 250B.C. that Archimedes discovered and recorded the principles of hydrostatics and flotation. Although the empirical understanding of hydrodynamics continued to improve with the development of fluid machinery, better sailing vessels and more intricate canal systems, the fundamental principles of classical hydrodynamics were not set forth until the 17^{th} and 18^{th} centuries. Newton, Daniel Bernoulli and Euler made the greatest contributions to establishing these principles.

Classical hydrodynamics, though a fascinating subject that appealed to mathematicians, was not applicable to many practical problems because the theory was based on inviscid fluids. The practicing engineers at that time needed design procedures that involved the flow of viscous fluids. Consequently, they developed empirical equations that were usable but narrow in scope. Thus, on the one hand, the mathematicians and physicists developed theories that in many cases could not be used by the engineers and on the other, the engineers used empirical equations that could not be used outside the limited range of application from which they were derived.

Near the beginning of the 20^{th} century, however, it was necessary to merge the general approach of physicists and mathematicians with the experimental approach of engineers to bring about significant advances in the understanding of flow processes. Osborne Reynolds' paper on 1883 on turbulence and later papers on the basic equations of motion contributed immeasurably to the development of fluid mechanics. After the turn of the century, Ludwig Prandtl proposed the concept of boundary layer. This concept not only paved the way to sophisticated analyses needed in the development of the airplane, but also resolved many of the paradoxes involved with the flow of a low-viscosity fluid.

1.3 Problem statement

The fluid flows have different conditions when it flows through a bend pipe (usually 45° and 90°). The flow may be a laminar or turbulent flow, depend on the length, diameter and configurations of the pipe. This can affect the pressure and the velocity of the fluids.

The problems that we are having here is the primary process in manufacturing computer hard disc which is the process of coating nickel sulphamate solution onto the disc. The nickel solution is normally fed into a mixing tank via a double L-bend pipe, which represents a sparger with small nozzle outlets. However, the coating process may be affected by non-uniform distribution of the nickel solution inside the mixing tank, resulting in an uneven nickel coating thickness at the disc surface. It is therefore desirable to have a uniform fluid discharge from the pipe nozzle outlets in order to obtain a uniform distribution of the nickel solution inside the mixing tank.

1.4 Objective and scope of the study

The scope of this study is aim to understand the basic and fundamental of fluid mechanics. The main objective of this study is;

I. To validate a previous computational fluid dynamics (CFD) result experimentally.

II. To estimate the fluid discharge from the present design of pipe nozzle outlets.

1.5 Summary

This chapter introduces the important to know how fluid flow and the applications that are connected in daily life. This chapter also introduces the problems faced to get the desirable fluid discharge from the nozzle. Hence, it is important to understand the mechanic of fluids.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

This literature review is based on some common equations and assumptions used to simplify a flow situation that are related to fluid properties especially in pipes or conduits. For example, under certain conditions, the viscosity can affect the flow significantly or can be neglected. Beside that, by having this literature review, it will refer to other people past gain and some of their experiment.

2.2 Internal flows

According to Schetz (1993), Ackeret (1967) describes four special features of internal flows;

1. The continuity equation plays a predominant role. The flow in a duct of any cross-sectional shape is modified by the growing boundary layer displacement thickness on the walls, which effectively narrows the channel and alters the pressure distribution and any inviscid flow in the central core. This interaction between the inviscid and viscous flows lead to an integral equation that must be solved. In long ducts, there is no clear distinction

between the boundary layer and the central core flow. All of these effects are more important for compressible flows.

2. Internal flows routinely have large deflections, such as in pipe bends, whereas large changes in angle are generally avoided in external flows. Thus, strong secondary flows, centrifugal forces and even other forces due to turning can be encountered much more frequently in internal than in external flows. Secondary flows are the flow components in a cross plane of a duct, such as in a pipe bend or the corner of a rectangular straight duct.
3. In internal flows, separation is usually followed quickly by reattachment, since the duct walls prevent the continued expansion of the free separation streamlines and wakes, unless the cross section of the duct is changing very rapidly. An analytical treatment of these phenomena is beyond the scope of this volume, but the practicing engineer must be aware of these effects.
4. Three-dimensional boundary layers occur in internal flows, as well as in external flows, but in internal flows, they often occur on rotating surfaces, such as disks and blade rotors.

One could add at least two other important characteristics of internal flows; first, great complexity in the flow patterns is routine; second, the occurrence of internal flows in practice is pervasive.

2.3 Bernoulli Equation

The Bernoulli equation is an approximate relation between pressure, velocity and elevation, and is valid in regions of steady, incompressible flow where net frictional forces are negligible (Çengel and Cimbala, 2006). Despite its simplicity, it has proven to be a very powerful tool in fluid mechanics. The Bernoulli equation was first stated in words by the Swiss mathematician Daniel Bernoulli (1700-1782) in a text written in 1738 when he was working in St. Petensburg, Russia. It was later derived in equation form by his associate Leonhard Euler in 1755.

Consider the motion of a fluid particle in a flow field in steady flow, applying Newton's second law (which is referred to as the conservation of linear momentum relation in fluid mechanics) in the s-direction on a particle moving along a streamline gives

$$\sum F_s = ma_s \tag{2.1}$$

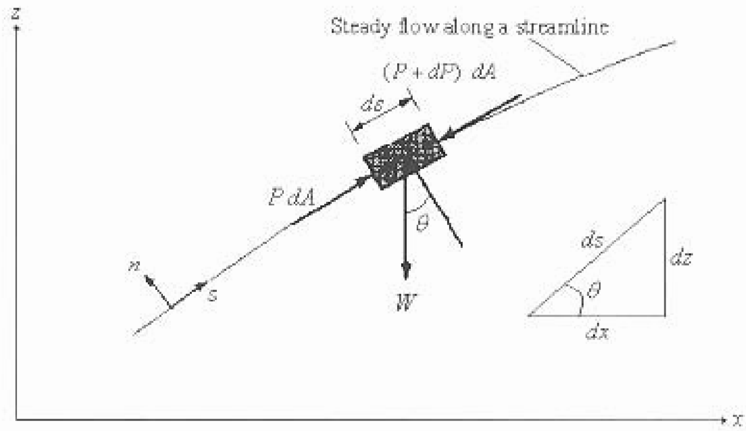

Figure 2.1 The forces acting on a fluid particle along a streamline.

(Courtesy of Çengel and Cimbala, 2006)

In region of flow where net frictional forces are negligible, the significant forces acting in the s-direction are the pressure (acting on both sides) and the component of the weight of the particle in the s-direction (Figure 2.1). Therefore, equation becomes

$$P\,dA - (P + dP)\ dA - W\sin\theta = mV\ \frac{dV}{ds} \tag{2.2}$$

where θ is the angle between the normal of the streamline and the vertical z-axis at that point. $m = \rho V = \rho\ dA$ is the mass, $W = mg = \rho g\ dA\ ds$ is the weight of the fluid particle, and $\sin\theta = \frac{dz}{ds}$. Substituting,

$$-\,dP\ dA - \rho g\ dA\ ds\ \frac{dz}{ds} = \rho\ dA\ ds\ V\ \frac{dV}{ds} \tag{2.3}$$

Canceling dA from each term and simplifying,

$$-\,dP - \rho g\ dz = \rho V\ dV \tag{2.4}$$

Noting that $V\ dV = \frac{1}{2}d(V^2)$ and dividing each term by ρ gives

$$\frac{dP}{\rho} + \frac{1}{2}d(V^2) + g\ dz = 0 \tag{2.5}$$

Integrating

Steady flow; $\int\frac{dP}{\rho} + \frac{V^2}{2} + gz = \text{constant (along a streamline)}$ (2.6) since the last two terms are exact differentials. In the case of incompressible flow, the first term also becomes an exact differential and its integration gives

$$\textit{Steady, incompressible flow;} \qquad \frac{P}{\rho}+\frac{V^2}{2}+gz=\text{constant} \qquad (2.7)$$

This is the Bernoulli equation, which is commonly used in fluid mechanics for steady, incompressible flow along a streamline in inviscid regions of flow. The value of the constant can be evaluated at any point on the streamline where the pressures, density velocity and elevation are known. The Bernoulli equation can also be written between any two points on the same streamline as

$$\textit{Steady, incompressible flow;} \quad \frac{P_1}{\rho}+\frac{V_1^2}{2}+gz_1=\frac{P_2}{\rho}+\frac{V_2^2}{2}+gz_2 \qquad (2.8)$$

The Bernoulli equation is obtained from the conservation of momentum for a fluid particle moving along a streamline. It can also be obtained from the first law of thermodynamics applied to a steady-flow system.

As recognized, $\frac{V^2}{2}$ as kinetic energy, gz as potential energy and $\frac{P}{\rho}$ as flow energy, all per unit mass. Therefore, the Bernoulli equation can be viewed as an expansion of mechanical energy balance and be stated as follows;

"The sum of the kinetic, potential and flow energies of a fluid particle is constant along a streamline during steady flow when the compressibility and frictional effects are negligible"

The kinetic, potential and flow energies are the mechanical forms of energy and the Bernoulli equation can be viewed as the conservation of mechanical energy principle. This is equivalent to the general conservation of energy principle for a system that does not involve any conversion of mechanical energy and thermal energy is conserved separately. The Bernoulli equation states that during steady, incompressible flow with negligible friction, the various forms of mechanical energy are converted to each other but the sum remains constant. In other words, there is no dissipation of mechanical energy to sensible thermal (internal) energy.

Recall that energy is transferred to a system as work when a force is applied to a system through a distance. In the light of Newton's second law of motion, the Bernoulli equation can also be viewed as; the work done by the pressure and gravity forces on the fluid particle is equal to the increase in the kinetic energy of the particle.

Despite the highly restrictive approximations used in its derivation, the Bernoulli equation is commonly used in practice since a variety of practical fluid flow problems can be analyzed to reasonable accuracy with it. This is because many flows of practical engineering interest are steady, compressibility effects are relatively small and net frictional forces are negligible in regions of interest in the flow.

2.4 Reynolds Number

A fluid flow may be broadly classified as either a viscous flow or an inviscid flow. An inviscid flow is one in which viscous effects do not significantly influence the flow and are thus neglected. In a viscous flow the effects of viscosity are important and cannot be ignored. Viscous flows include the broad class of internal flows, such as flows in pipes, conduits and in open-channels. A viscous flow can be classified as either laminar flow or turbulent flow.

According to Çengel and Cimbala (2006), a careful inspection of flow in a pipe reveals that the fluid flow is streamlined at low velocities but turns chaotic as the velocity is increased above a critical value, as shown in Figure 2.2. The flow regime in the first case is said to be laminar, characterized by smooth streamlines and highly ordered motion, and turbulent in the second case, where it is characterized by velocity fluctuations and highly disordered motion. The transition (intermittent) from laminar to turbulent flow does not occur suddenly; rather, it occurs over some region in which the flow fluctuates between laminar and turbulent flows before it becomes fully turbulent.

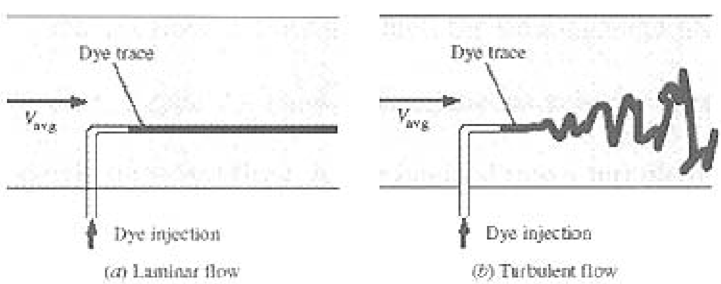

Figure 2.2 The behavior of colored fluid injected into the flow in laminar and turbulent flows in a pipe.

(Courtesy of Çengel and Cimbala, 2006)

According to Potter and Wiggert (1997), in a laminar flow the fluid flows with no significant mixing of neighboring fluid particles. If dye were injected into the flow, it would not mix with the neighboring fluid except by molecular activity; it would retain its identity for a relatively long period of time. Viscous shear stresses always influence a laminar flow. The flow may be highly time dependent, as shown by the output of a velocity probe in Figure 2.3 (a), or it may be steady, as shown in Figure 2.3 (b).

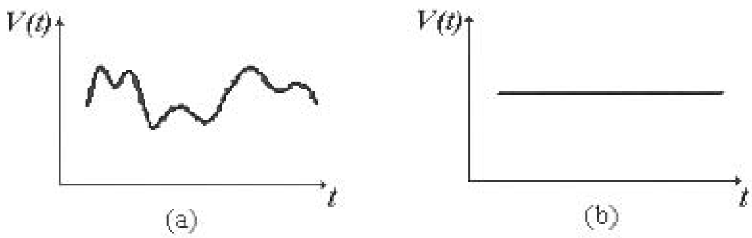

Figure 2.3 Velocity as a function of time in a laminar flow:

(a) unsteady flow; (b) steady flow

(Courtesy of Potter and Wiggert, 1997)

In a turbulent flow fluid motions vary irregularly so that quantities such as velocity and pressure show a random variation with time and space coordinates. The physical quantities are often described by statistical averages. In this sense we can define a steady turbulent flow: a flow in which the time-average physical quantities do not change in time. Figure 2.4 shows instantaneous velocity measurements in an unsteady and a steady turbulent flow. A dye injected into a turbulent flow would mix immediately by the action of the randomly moving fluid particles; it would quickly lose its identity in this diffusion process.

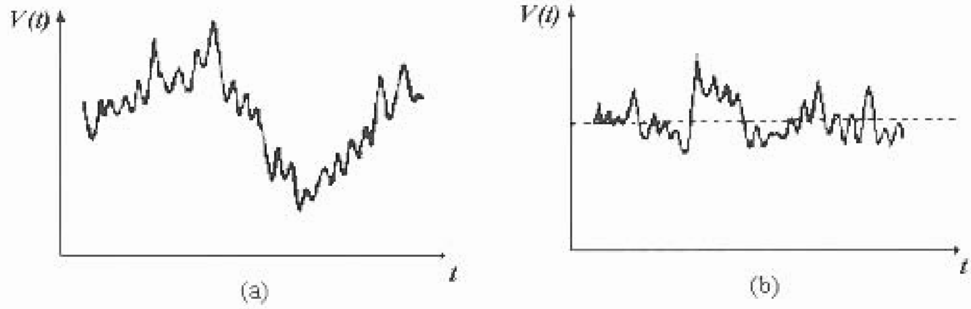

Figure 2.4 Velocity as a function of time in a turbulent flow;
(a) unsteady flow; (b) steady flow
(Courtesy of Potter and Wiggert, 1997)

According to Roberson and Crowe (1997), turbulent flow is characterized by a mixing action throughout the flow field and this mixing is caused by eddies of varying size within the flow. Simple observations will reveal this type of flow in rivers and in the atmosphere. Gusts of wind are the result of large-scale eddies that at times reinforce and at other times subtract from the mean wind velocity. Laminar flow, on the other hand, is devoid of the intense mixing phenomena and eddies common to turbulent flow. Thus this flow has a very smooth appearance. A typical example is the flow of honey or thick syrup from a pitcher.

The existence of these laminar, transitional and turbulent flow regimes can be verify by injecting some dye streaks into the flow in a glass pipe, as the British engineer Osborne Reynolds (1842-1912) did over a century ago. The dye streak forms a straight and smooth line at low velocities when the flow is laminar, has bursts of fluctuations in the transitional regime and zigzags rapidly and randomly when the flow becomes fully turbulent can be observed. These zigzags and the dispersion of the dye are indicative of the fluctuations in the main flow and the rapid mixing of fluid particles from adjacent layers.

The intense mixing of the fluid in the turbulent flow as a result of rapid fluctuations enhances momentum transfer between fluid particles, which increases the friction force on the surface and thus the required pumping power. The friction factor reaches a maximum when the flow becomes fully turbulent.

The transition from laminar to turbulent flow depends on the geometry, surface roughness, flow velocity, surface temperature and type of fluid, among other things. After exhaustive experiments in the 1880s, Osborne Reynolds discovered that the flow regime mainly on the ratio of inertial forces to viscous forces in the fluid. This ratio is called the Reynolds number and is expressed for internal flow in a circular pipe.

$$\mathrm{Re} = \frac{inertial\ foces}{viscous\ forces} = \frac{V_{avg} D}{v} = \frac{\rho V_{avg} D}{\mu} \tag{2.9}$$

where V_{avg} = average flow velocity (m/s)

D = characteristic length of the geometry (diameter in this case, in m)

$v = \frac{\mu}{\rho}$ = kinematic viscosity of the fluid (m^2/s).

Note that the Reynolds number is a dimensionless quantity. Also, kinematic viscosity has the unit m^2/s and can be viewed as viscous diffusivity or diffusivity for momentum. The Reynolds number is probably the most used of all the dimensionless parameters in fluid mechanics, since it is the most descriptive parameter for distinguishing the nature flows of engineering interest (Graebel, 2001).

In certainly is desirable to have precise values of Reynolds numbers for laminar, transitional and turbulent flows, but this is not the case in practice. It turns out that the transition from laminar to turbulent flow also depends on the degree of disturbance of the flow by surface roughness, pipe vibration and fluctuations in the flow. Under most practical conditions, the flow in a circular pipe:

$Re \leq 2300$	laminar flow
$2300 \leq Re \leq 4000$	transitional flow
$Re \geq 4000$	turbulent flow

2.5 Pitot Tubes

Pitot tubes (also called Pitot probes) and Pitot-static tubes are widely used for flow rate measurement. It is named after Henri de Pitot (1695-1771), a French hydraulic engineer of the eighteenth century. Pitot's idea was an important contribution to flow measurement, even though his analysis of the theory of his device was in error (Graebel, 2001). A Pitot tube is just a tube with a pressure, while a Pitot-static tube has both a stagnation pressure tap and several circumferential static pressure taps and it measures both stagnation and static pressures. Pitot was the first person to measure velocity with the upstream pointed tube, while French engineer Henry Darcy (1803-1858) developed most of the features of the instruments that are use nowadays, including the use of small openings and the placement of the static tube on the same assembly. Therefore, it is more appropriate to call the Pitot-static probes Pitot-Darcy probes (Çengel and Cimbala, 2006).

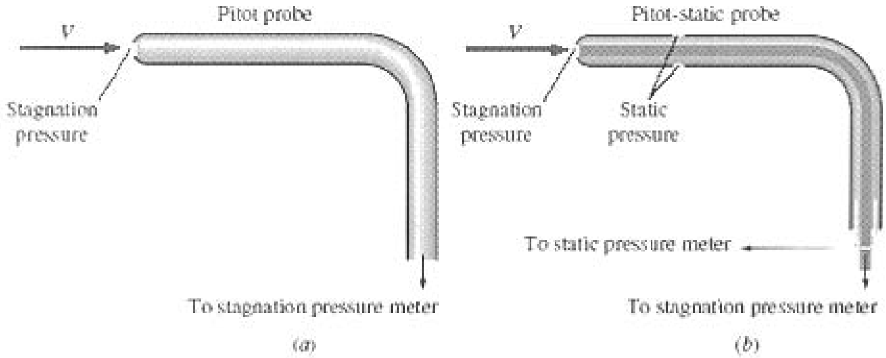

Figure 2.5 (a) A Pitot probe measures stagnation pressure at the nose of the probe, while (b) a Pitot-static probe measures both stagnation pressure and static pressure, from which the flow can be calculated.

(Courtesy of Çengel and Cimbala, 2006)

The Pitot-static tube measures local velocity by measuring the pressure difference in conjunction with the Bernoulli equation. It consists of a slender double-tube aligned with the flow and connected to a differential pressure meter. The inner tube is fully open to flow at the nose, and thus it measures the stagnation pressure at the location. The outer tube is sealed at the nose, but it has holes on the side of the outer wall and thus it measures the static pressure. For incompressible flow with sufficiently high velocities, the Bernoulli equation is applicable and can be expressed as

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \qquad (2.10)$$

Noting as $z_1 \cong z_2$ since the static pressure holes of the Pitot-static tube are arranged circumferentially around the tube and $V_1 = 0$ because of the stagnation conditions, the flow velocity $V = V_2$ becomes

$$V = \sqrt{\frac{2(P_1 - P_2)}{\rho}} \tag{2.11}$$

which is known as the Pitot formula. If the velocity is measured at a location where the local velocity is equal to the average flow velocity, the volume flow rate can be determined.

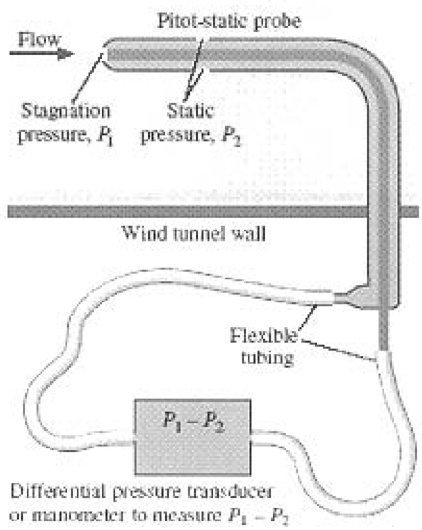

Figure 2.6 Measuring flow velocity with a Pitot-static probe.

(Courtesy of Çengel and Cimbala, 2006)

The Pitot-static tube is a simple, inexpensive and highly reliable device since it has no moving parts. It also causes very small pressure drop and usually does not disturb the flow appreciably. However, it is important that it be properly aligned with the flow to avoid significant errors that may caused by misalignment (Çengel and Cimbala. 2006). They are relatively insensitive to misalignment as long as they are aligned within $\pm 15^{\circ}$ of the flow direction (Graebel, 2001). If the flow is not parallel to the probe head, the measurement error is about 1% at a yaw angle of 5° and if the yaw angle is greater than 5° the measurement error may be substantial (Potter and Wiggert, 1997).

According to Graebel (2006), the pitot tubes introduce little if any energy loss in the flow. The can have difficulties with fouling in flows with the suspended solids, which can plug the openings. In cases where the pitot tube is used in a liquid, great care must be exercised to eliminate all traces of gas bubbles in the connecting lines attached to the pitot tube.

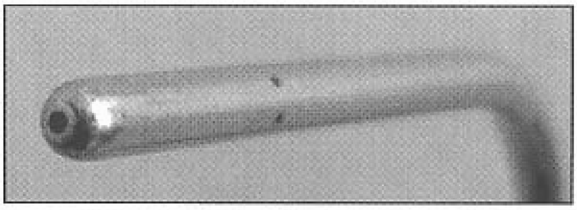

Figure 2.7 Close-up of a Pitot-static probe, showing the stagnation pressure hole and two of the five static circumferential pressure holes.
(Courtesy of Çengel and Cimbala, 2006)

2.6 Fluid Flow Simulation

A fluid flow simulation in a double L-bend pipe with small nozzle outlets had been done by Rigit et al. (2005). The flow simulation was performed with a commercially available computational fluid dynamics (CFD) package, Star-CD. The effects of the L-bend and small nozzle outlets on the velocity and pressure distributions in the pipe were discussed in the research.

The CFD explores the effects of a double 90°C L-bend and a series of equally spaced small nozzles along one side of the pipe horizontal arm on the fluid flow. The configuration used in the study is shown in Figure 2.8. The pipe geometry is a double L-bend configuration with both perpendiculars to each other. The length of the smooth pipe is modeled to be equivalent to 20 times the diameter of the pipe. This double L-bend configuration also replicated many pipe structures present in a range of pipe networks.

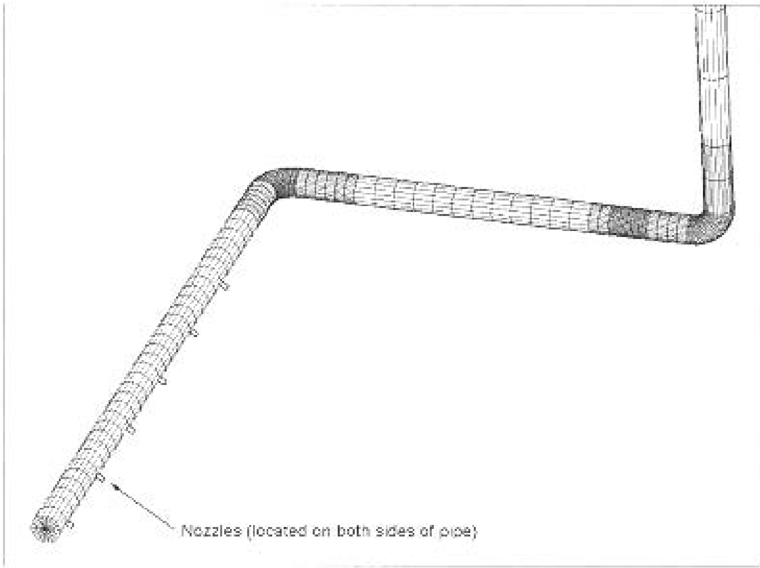

Figure 2.8 The configuration of the double L-bend pipe with series of nozzles located after the 2nd bend.

(Courtesy of Rigit et al, 2005)

The periodic cross-sectional contour-plots of the velocity-magnitude within the 2nd L-bend of the sparger is shown in Figure 2.9. It is observed at the middle-part of the pipe, the plots tend to exhibit horizontal-forms of contour-plots. This signifies the fluid-flow interaction around the boundary-regions of the nozzles.

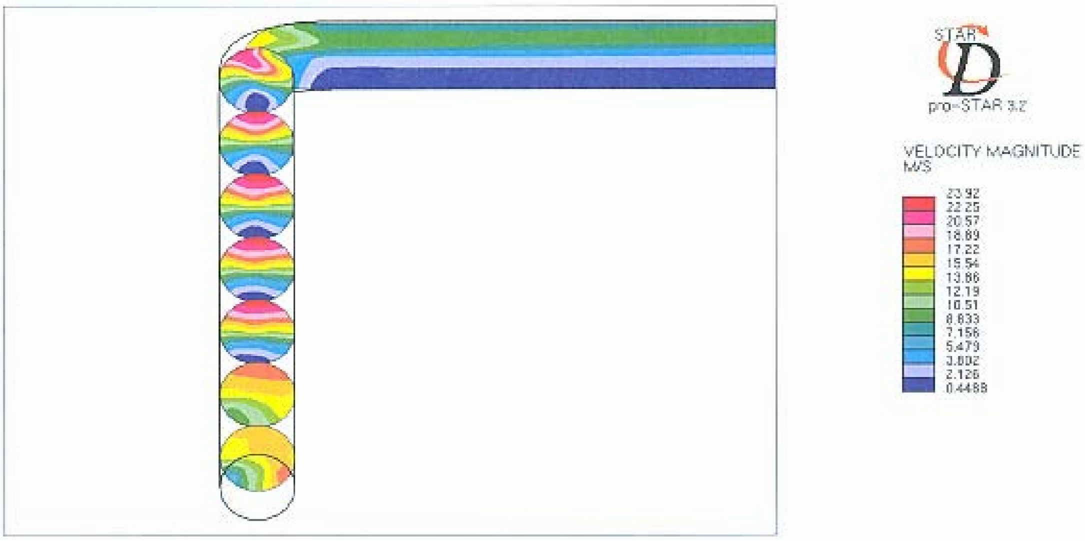

Figure 2.9 Periodic cross-sectional velocity-magnitude in contour-plot.

(Courtesy of Rigit et al, 2005)

Figure 2.10 shows the particle-tracking plot with ribbon-widths of 0.1 models units and twist-magnification of 1. The ribbons are also plotted with mesh-plotting. This plot allows a much closer observation on the fluid-flow within the sparger at the 2nd L-bend. It is observed that the fluid flow is experiencing a vortex flow.

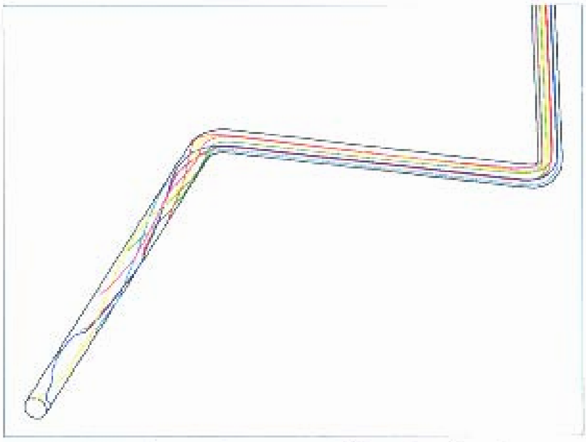

Figure 2.10 Particle-tracking ribbon-form at 2nd L-bend of sparger in isometric view.

(Courtesy of Rigit et al, 2005)

Figure 2.11 and Figure 2.12 depicted the velocity and total pressure magnitude respectively for the whole pipe configuration. The fluid flow at the 1st L-bend showing adverse pressure gradient indicating the occurrence of wake. However, downstream of the pipe before the 2nd L-bend, the flow settles and the distribution shows that the flow tends to be uniform. On the other hand, this flow is not observed after the 2nd L-bend.

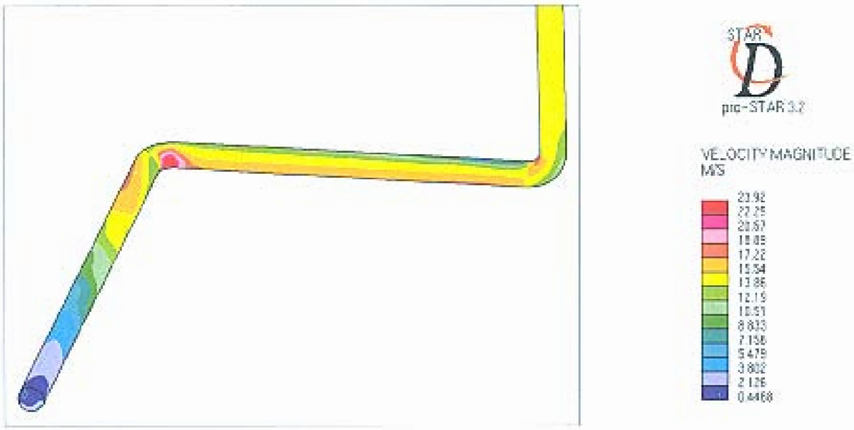

Figure 2.11 The velocity distribution in the pipe.

(Courtesy of Rigit et al, 2005)

Figure 2.12 The total pressure distribution for the pipe.

(Courtesy of Rigit et al, 2005)

This is clearly shown in Figure 2.13, where 20 contour-plots of the velocity magnitude at this section of the pipe are presented, that the flow is circulating.

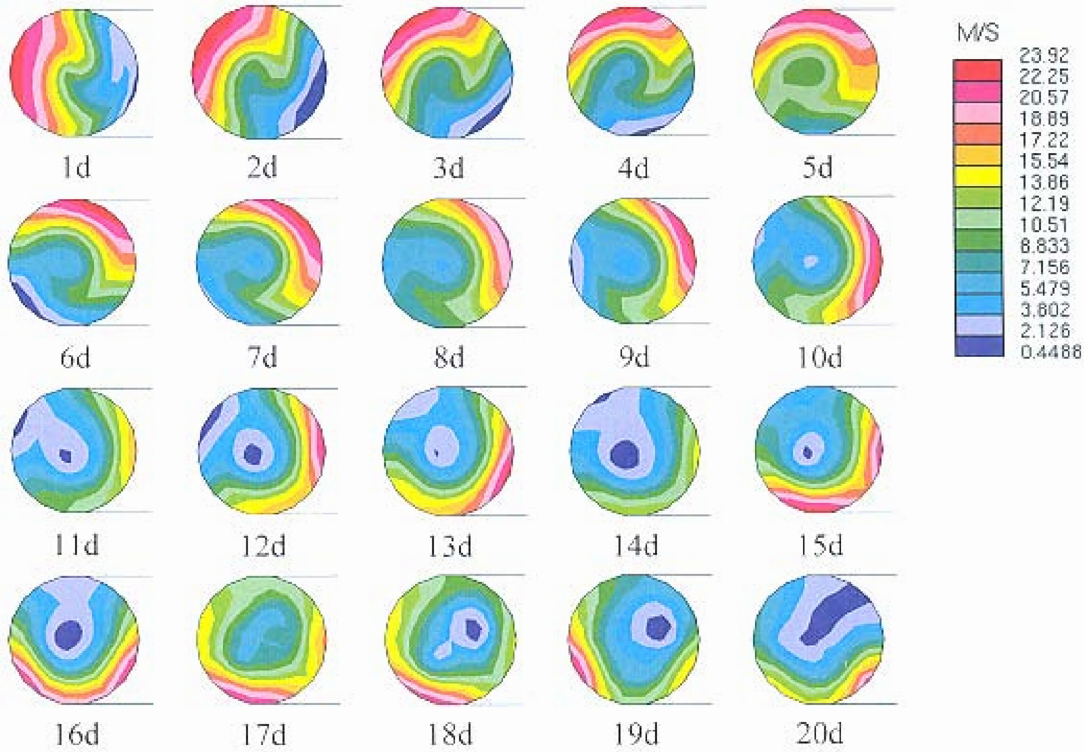

Figure 2.13 The 20 contour-plots of the velocity magnitude at cross-sectional view. (Courtesy of Rigit et al, 2005)

2.7 Summary

This chapter provides the basic knowledge of fluid flow in the pipe, the flow characteristic and how to calculate the flow rate, the pressure and the velocity of the fluid in the pipe. It also showed the result of fluid flow simulation using computational fluid dynamics.

CHAPTER 3

METHODOLOGY OF RESEARCH

3.1 Introduction

The computational fluid dynamics (CFD) explores the effects of a double 90° L-bend and series of equally spaced small nozzles along one side of the pipe horizontal arm on the fluid flow (Rigit et al, 2005). Objective of this study are to validate the CFD result experimentally and to calculate the fluid discharge from the present design of pipe nozzle outlets.

3.2 Calibration of the flow meter

Flow measurement is very important in industries handling fluids in their several processes. The fluid flow rate determination has many repercussions in the economy and life an enterprise, because its market presence could be affected for the low quality of its products or for not to give a good attention to the clients due to a bad measurement practice of the flow rate (Silva *et al.* 1997).

Thus, calibration is necessary in order to achieve accurate and uniform applications. In this case, the flow meter is being calibrated to determine the flow meter accuracy and efficiency.

The procedures for the calibration process are;

1. Firstly, mark the levels as t_o and t_1 as shown in Figure 3.1. The different between t_o and t_1 is set to h = 15cm in height.
2. Then, fill up the tank 1 with water until the water level is slightly above from t_o (Figure 3.1).
3. The flow rate of the flowmeter is adjusted to 20LPM (liter per minute) by the control valve.
4. When the pump is switch on, the time will be taken when it reaches t_o and stop when it reaches t_1. This reading will be taken at least 3 times for each of the flow rate.
5. Steps 4 will be repeated with different flow rate (30LPM, 40LPM, 50 LPM and 60LPM).
6. The average time of each reading will be use to determine the actual flow rate.

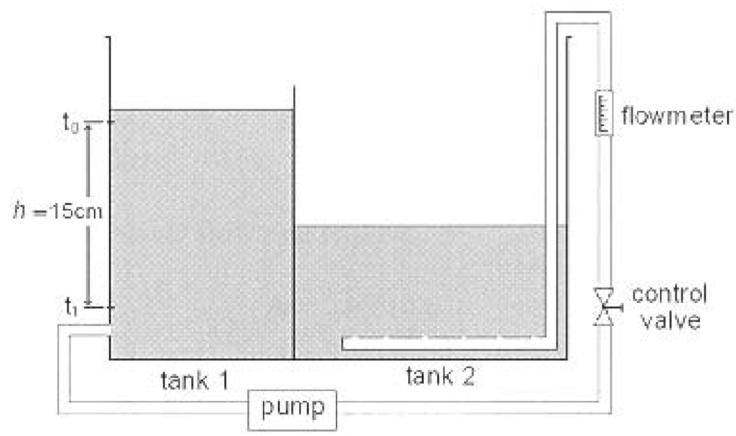

Figure 3.1 The schematic diagram for the calibration of the flowmeter.

The volume of water that flow out from tank 1 in t second is

$$\mathbf{V} = Ah \tag{3.1}$$

where $A = (0.487) \times (0.197)$ m and $h = 0.15$ m

$= 0.096$ m²

and the volume, $\mathbf{V} = (0.096) \times (0.15)$

$= 0.0144$ m³

The flow rate,
$$Q = AV \tag{3.2}$$

where A is the area (diameter of the pipe) and V is the velocity (in the pipe). And the flow rate, Q can also be

$$Q = \frac{\mathbf{V}}{t} \tag{3.3}$$

where $\mathbf{V}$ is the volume amount and t is the time. In this case, the volume is the area of tank 1 with the height of 15cm and t is the time taken by the water level from t_0 to t_1 .

For example;

The volume, $\mathbf{V} = 0.0144$ m³

Time taken, t = 15s

So, the actual flow rate will be,

$$Q = \frac{\mathbf{V}}{t} = \frac{0.0144}{15}$$

$$= 0.00096 \text{ m}^3/\text{s}$$

3.3 Pressure in the pipe

The fluid discharge from the every of the nozzles are not the same because of the difference velocity throughout the pipe. The average velocity in the pipe can be calculated if the pressure in the pipe can be determined. A pitot-static tube can be used to determine the pressure in the pipe.

The apparatus for the experiment (Figure 3.2) are the Pitot-static tube, a plastic tube (D=5mm, L=2m), a pair of steel wire (L≈50cm) and a wood block (7x8x9cm). The wood block is for the Pitot-static tube to stand so it would not move during the experiment (Figure 3.3). The Pitot-static tube must be at the centre in the pipe and parallel with the flow of the water (Figure 3.4). Any misalignment will make an error and affect the reading of the pressure. This problem has been discussed in the literature review.

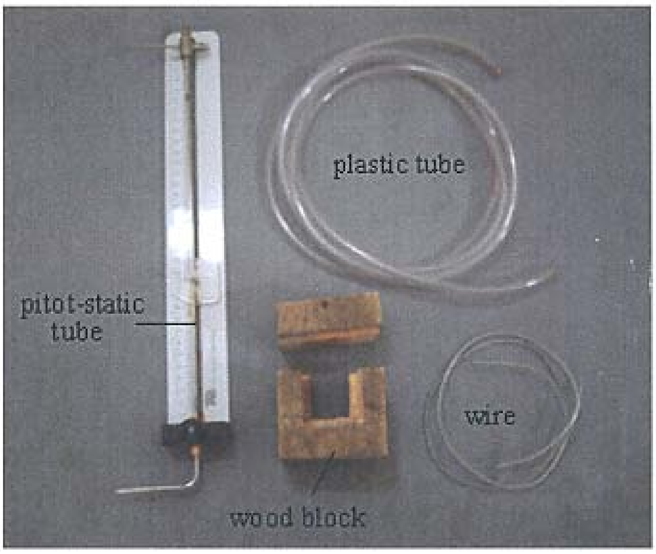

Figure 3.2 The apparatus to find the pressure in the pipe.

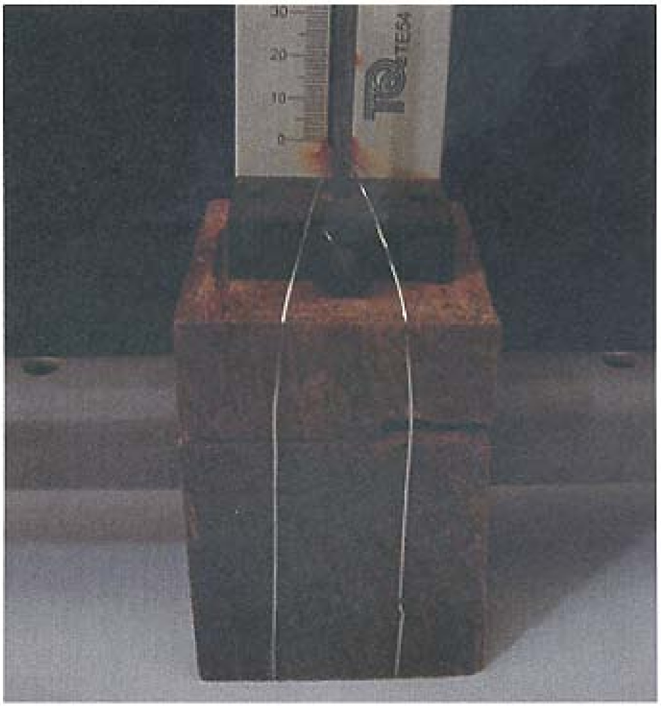

Figure 3.3 Pitot-static tube is hold using the wood block.

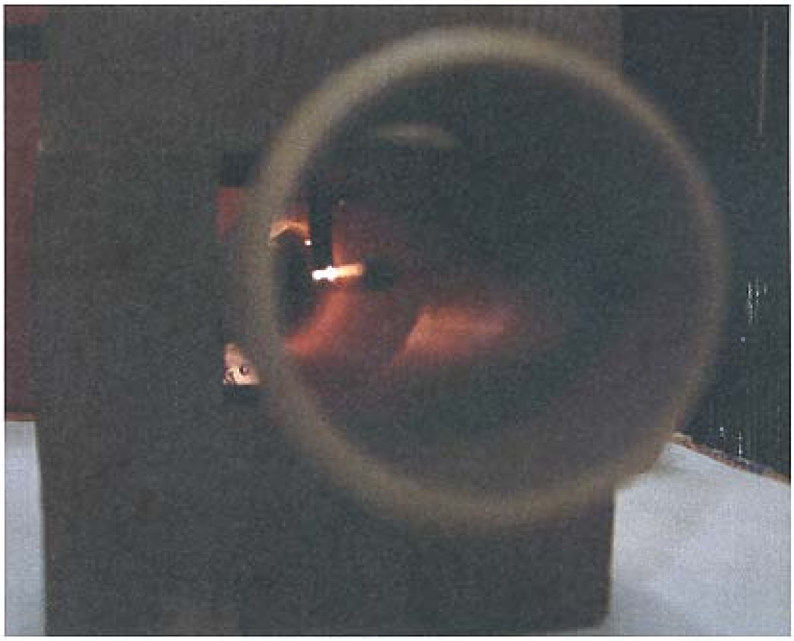

Figure 3.4 The pitot-static tube is at the centre of the pipe.

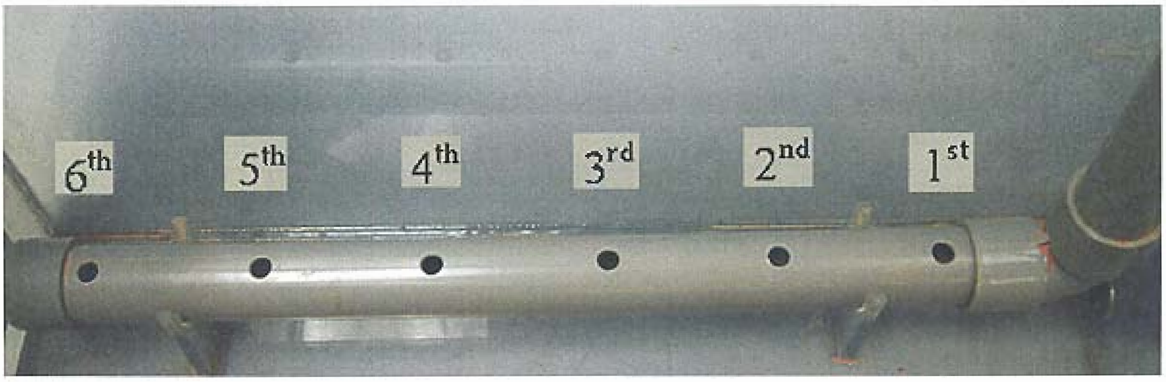

Figure 3.5 The nozzles (from 1^{st} till 6^{th} hole) of the pipe.

The experiment has been carried out accordingly and the procedures to measure the pressure using Pitot-static tube for the experiment are;

1. The Pitot-static tube will be placed in the 2^{nd} nozzle of the pipe. The 1^{st} nozzle will not be counted because the L-shape of the Pitot-static tube will be at bend of the pipe (Figure 3.5).
2. The flow rate of the flowmeter is adjusted to 30LPM (liter per minute) by the control valve when the pump is switch on.
3. The height difference of water in the plastic tube will be taken when the water level in the plastic tube is stable and constant. This reading will be taken for 3 times.
4. After that, the flow meter will be adjusted to 40LPM and step 3 will be repeated.
5. Step 2, 3 and 4 will be repeated for the next nozzle (3^{rd}, 4^{th}, 5^{th} and 6^{th} hole).
6. The average time of each reading will be use in the data calculation.

3.4 Flow rate from each of the nozzle.

The flow rate that come out from each of the nozzle can be measured by using a plastic tube (Outer diameter=9mm) and a bottle (**V**=1.5L). The time, t for the water to fill up the bottle can be used to calculate the flow rate. The procedures to measure the flow rate are;

1. The plastic tube will be placed in the 1st nozzle as shown in Figure 3.6 and the bottle will be mark as shown in Figure 3.7 which is to show the water has reach 1.5L.
2. The flow rate of the flowmeter is adjusted at 20LPM by using the control valve.
3. The time will be taken when the water starts to fill up the bottle and stop when the water reaches the marked level. This reading will be taken at least 3 times for each of the flow rate (20LPM, 30LPM and 40LPM).
4. Step 2 and 3 will be repeated for the next nozzle (2nd, 3rd, 4th, 5th and 6th hole).
5. The average time of each reading will be use to determine flow rate from each of the nozzle.

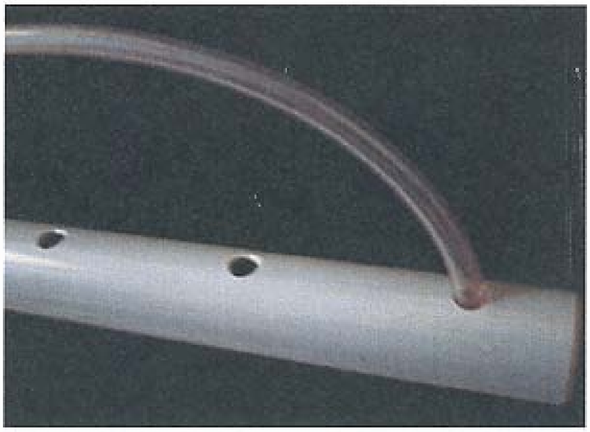

Figure 3.6 The plastic tube is placed in the 1st nozzle.

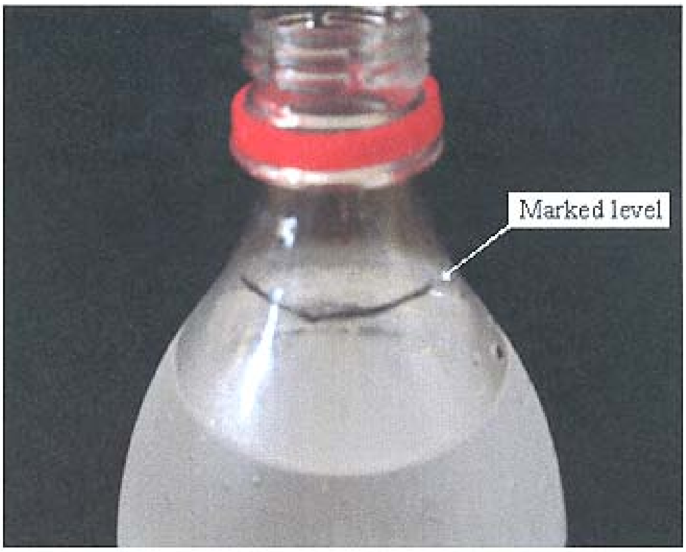

Figure 3.7 The marked level indicates 1.5L.

3.5 Summary

This chapter includes the data collection method and analysis. The results of this experiment will be used to compare with the CFD results and is discussed in the next chapter.

CHAPTER 4

RESULTS & DISCUSSION

4.1 Introduction

In this chapter, the data and result from the experiment that has been done will be use in the calculation. Furthermore, it will be use to compare with the CFD result and will be discussed later.

4.2 Calibration of the flowmeter

The result for the calibration is shown in Table 4.1. The time has been taken 3 times and the average time will be use to determine the actual flow rate in the tank. The time taken is different for every flow rate that indicates in the flowmeter.

Table 4.1 The time taken from each of the flowmeter reading.

t_1	t_2	t_3	t_{avg} (s)	flowmeter (LPM)
24.01	24.00	24.00	24.00	20
17.95	17.80	17.86	17.87	30
14.60	14.75	14.50	14.62	40
11.91	12.10	11.81	11.94	50
9.99	9.76	9.90	9.88	60

Using equation 3.3, $Q = \frac{V}{t}$, so the actual flow rate that flows in the pipe from tank 1 to tank 2 using the pump is shown in Table 4.2.

Table 4.2 The actual flow rate from the flowmeter reading.

actual flowrate (m^3/min)	flowmeter reading (m^3/min)
0.036	0.02
0.048	0.03
0.059	0.04
0.070	0.05
0.087	0.06

From this data, a graph of the actual flow rate vs. flowmeter reading can be represented in Figure 4.3. From the flowmeter reading, the actual flow rate in the pipe that is required in the calculation can be determined from the graph.

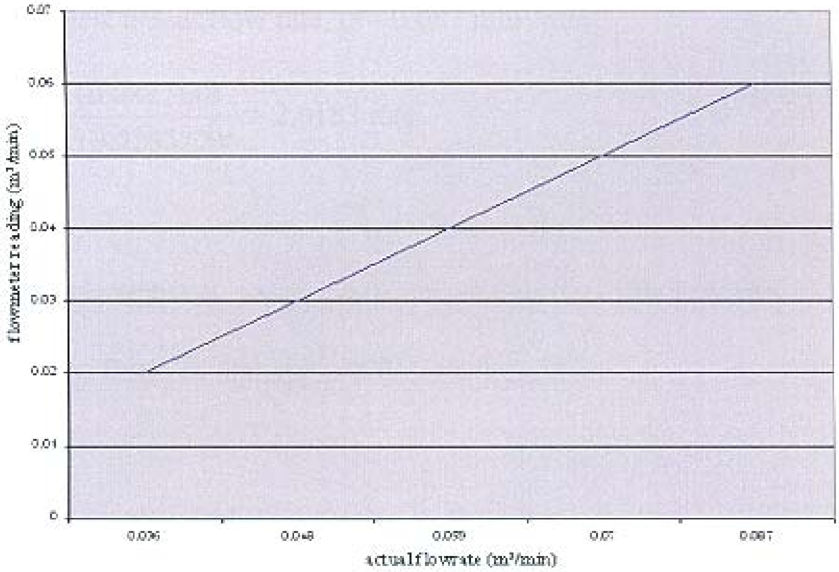

Figure 4.3 The actual flow rate vs flowmeter reading.

4.3 Characteristic of the flow

The characteristic of the flow in the pipe can be determined using the Reynolds number equation (equation 2.9). But before that, the average velocity in the pipe has to be determined first. Using equation 3.2, the actual flow rate and the diameter of the pipe are known, and then the velocity in the pipe can be determined.

Flow rate, $Q = AV$ Reynolds number, $\mathrm{Re} = \dfrac{\rho V D}{\mu}$

Diameter of the pipe, D = 0.0285m

Water at 20°C, $\rho = 998\text{kg/m}^3$ and $\mu = 1.002 \times 10^{-3}\text{kg/m.s}$

Using the lowest actual flow rate, Q= 0.036 min³/min

$$V = \frac{(0.036/60)}{\pi(0.0285/2)^2} = 1.0092 \text{ m/s}$$

so, Re = 29149.97, which this is a turbulent flow (Re ≥ 4000).

Using the largest actual flow rate, Q= 0.087 min³/min

$$V = \frac{(0.087/60)}{\pi(0.0285/2)^2} = 2.0183 \text{ m/s}$$

so, Re = 58297.0, which this is a turbulent flow (Re ≥ 4000).

As a result, the flows in the pipe are all turbulent flow.

4.4 The pressure and the velocity in the pipe.

The pressure in the pipe can be used to determine the average velocity throughout the pipe using the Pitot formula (equation 2.11). But, it is necessary to know how to measure the pressure in the pipe. The pressure in the pipe will make some difference the water level in the tube (Figure 4.4). This difference in height, x will be used to determine the velocity in the pipe.

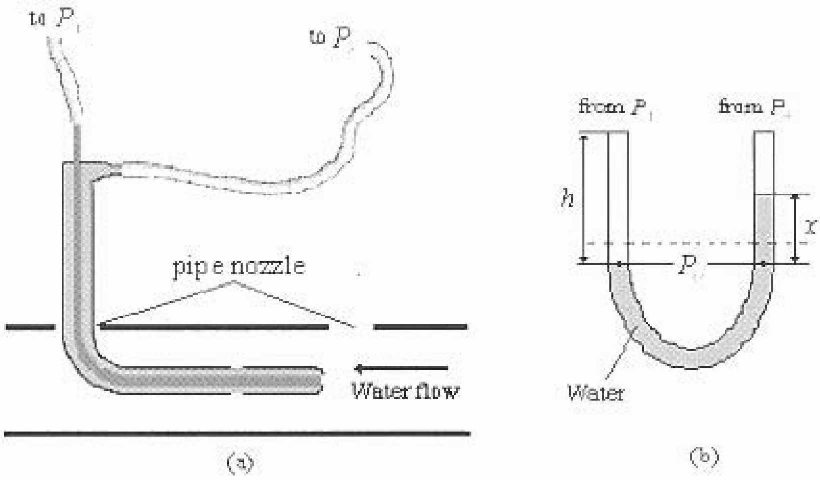

Figure 4.4 (a) the Pitot-static tube in the pipe,

(b) the tube with water connected with the Pitot-static tube in (a).

Table 4.5 The height difference, x in the tube for each of the nozzle.

2nd nozzle

Flowmeter	height difference, x (m)			
	1	2	3	average
30 LPM	0.083	0.087	0.087	0.086
40 LPM	0.144	0.144	0.143	0.144

3rd nozzle

Flowmeter	height difference, x (m)			
	1	2	3	average
30 LPM	0.058	0.056	0.059	0.058
40 LPM	0.110	0.106	0.111	0.109

4th nozzle

Flowmeter	height difference, x (m)			
	1	2	3	average
30 LPM	0.011	0.009	0.012	0.011
40 LPM	0.035	0.038	0.037	0.037

5th nozzle

Flowmeter	height difference, x (m)			
	1	2	3	average
30 LPM	0.001	0.002	0.001	0.001
40 LPM	0.005	0.009	0.004	0.006

6th nozzle

Flowmeter	height difference, x (m)			
	1	2	3	average
30 LPM	0.0	0.0	0.0	0.0
40 LPM	0.002	0.002	0.003	0.002

From Figure 4.4(b),

the left hand side, $P_Q = P_A + \rho_{air} gh$

the right hand side, $P_Q = P_B + \rho_{air} g(h-x) + \rho_{water} gx$

so, $P_A - P_B = \rho_{air} gh - \rho_{air} gx + \rho_{water} gx - \rho_{air} gh$

$$P_A - P_B = \rho_{water} gx - \rho_{air} gx \tag{4.1}$$

Using Pitot formula, $V = \sqrt{\dfrac{2(P_1 - P_2)}{\rho}}$, where $P_1 = P_A$ and $P_2 = P_B$ and $\rho = \rho_{water}$

$$V = \sqrt{\frac{2(P_A - P_B}{\rho_{water}}} = \sqrt{\frac{2(\rho_{water} - \rho_{air})gx}{\rho_{water}}}$$

$$V = \sqrt{(1 - \frac{\rho_{air}}{\rho_{water}})2gx} \tag{4.2}$$

at 20°C, $\rho_{air} = 1.2\text{kg/m}^3$, $\rho_{water} = 998\text{kg/m}^3$, $g = 9.81\text{m/s}^2$

Table 4.6 The average velocity at the centre of the pipe.

Nozzle	Average velocity (m/s) at	
	30LPM	40LPM
2^{nd}	1.3	1.68
3^{rd}	1.07	1.46
4^{th}	0.46	0.85
5^{th}	0.14	0.34
6^{th}	0	0.2

4.5 Flow rate from each of the nozzle

The experiment to measure the flow rate that comes out from each of the nozzle has been carried out. The time taken to fill up the 1.5L bottle is shown in Table 4.7. The average time will be use to calculate the flow rate from each of the nozzle.

Table 4.7 The time taken (in second) to fill up the 1.5L bottle.

1^{st} nozzle

Flowmeter (LPM)	t_1	t_2	t_3	t_{avg}
20	74.16	73.94	75.29	74.46
30	60.50	60.59	61.54	60.88
40	52.23	52.32	51.98	52.18

2^{nd} nozzle

Flowmeter (LPM)	t_1	t_2	t_3	t_{avg}
20	69.21	68.76	68.75	68.91
30	59.24	59.34	59.05	59.21
40	48.69	49.35	48.78	48.94

3^{rd} nozzle

Flowmeter (LPM)	t_1	t_2	t_3	t_{avg}
20	66.70	66.93	66.36	66.66
30	56.65	56.36	56.65	56.55
40	47.15	46.88	47.21	47.08

4^{th} nozzle

Flowmeter (LPM)	t_1	t_2	t_3	t_{avg}
20	65.06	65.35	65.42	65.28
30	55.37	55.70	54.55	55.21
40	46.15	45.25	45.32	45.57

5^{th} nozzle

Flowmeter (LPM)	t_1	t_2	t_3	t_{avg}
20	62.99	63.08	62.50	62.86
30	52.81	52.82	53.04	52.98
40	44.08	44.24	44.16	44.16

6th nozzle

Flowmeter (LPM)	t_1	t_2	t_3	t_{avg}
20	61.26	61.08	60.35	60.90
30	51.38	50.70	50.79	50.96
40	42.83	42.35	42.17	42.45

Using the equation 3.3, $Q = \frac{\mathbf{V}}{t}$, the flow rate from each of the nozzle can be determined.

Where, $\mathbf{V}$ = 1.5L = 0.0015m³. So the flow rate for each of the nozzle at 20LPM, 30LPM and 40LPM is shown in Table 4.8.

Table 4.8 The flow rate (m³/min) from each of the nozzle.

Nozzle	Flow rate (m³/min) at		
	20LPM	30LPM	40LPM
1st	0.00121	0.00148	0.00172
2nd	0.00131	0.00152	0.00184
3rd	0.00135	0.00159	0.00191
4th	0.00138	0.00163	0.00197
5th	0.00143	0.00170	0.00204
6th	0.00148	0.00177	0.00212

4.6 Discussion

From the experiment that have been done, the data and the results have been calculated and determined using some of the equation that have been discussed earlier. An assumption has been made during the experiment and calculation that the minor losses in the experiment are neglected. As been done earlier, the characteristic of the flow in the pipe is turbulent. This is because of the high flow rate and velocity that flows in the pipe. The velocity of the fluid flow at the centre of the pipe is decreasing from the last bend of the pipe until the end of the pipe (Table 4.6). The flow rate at 20LPM is not been mentioned because of the low pressure in the pipe where there is almost no different in height level when using Pitot-static tube. The decreasing of the velocity in the pipe is also happened in the CFD simulation results where shown in Figure 2.11. The velocity distribution in the pipe becomes slow at the end of the pipe. But the flow rate that discharge and comes out from each of the nozzle is different. The flow rate from the first nozzle is increasing till the last (6^{th}) nozzle. From the flow rate equation, $Q = AV$, the velocity from the first nozzle till the last nozzle should also be increasing. This is because the existing of secondary flows in the pipe.

The secondary flows of a pipe can be described as the cross-stream motion of the primary phase across the mainstream of the flow, usually resulting from dynamic pressure differences created by a bend (Aroussi et al, 2004). According to Sánchez Silva et. al. (1997), when a flow, initially in a straight pipe finds a change in direction, the particles with grater velocity near the central axis are subjected to a greater centrifugal force than the particles with low velocity near the walls. The centrifugal force provokes a superposition in primary axial flow of a transversal movement known as secondary flow, in which the flow from the central region of the duct moves out of the center and the particles near the wall moves toward the center. According to Rigit et. al. (2005), the results from the CFD simulations show that there exists a secondary flow in both bends, which due to the dynamic pressure differences created on each bend.

From the Figure 2.13, the contour-plots of the velocity magnitude at cross-sectional view showed only at the center of the pipe the velocity is decreasing and at the side of the pipe, the velocity is still high. The nozzle is placed at the side of the pipe and this is why the flow rate should also be higher. The last nozzle has higher flow rate (Table 4.8) and decrease till the first nozzle because it is at the end of the pipe and the fluids tends to go out through it. In order to address this non-uniformity of the fluid discharge from the nozzles, the location of the nozzle along the pipe and the size of the nozzle diameter could be varied as a function of the pipe length.

4.7 Summary

The calculation of the flow rate, the pressure and the velocity in the pipe and through out the pipe was carried out. From the calculation, the experiment result can be compared with the CFD result.

CHAPTER 5

CONCLUSION & RECOMMENDATION

5.1 Conclusion

This study has demonstrated experimentally how to find and calculate the flow rate, pressure and velocity of the fluid flow in the pipe and discharge from the nozzles. The results from the experiment and calculation have been compared with the computational fluid dynamics (CFD) result. It showed that the CFD result is valid because of the velocity distribution in the pipe is same with the experiment result. It also includes that the secondary flows exists in the pipe. The various configurations produce difference secondary flow patterns.

In order to obtain a uniform fluid discharge from the nozzles, the fluid flow past the second bend must be controlled so that the secondary flows will not occur. This suggests that the pipe configuration and design needs to be improved perhaps by including guided vanes on the L-bends and adjusting the position of the nozzle and the diameter along the pipe.

5.2 Recommendation

In the future, better understanding and knowledge of the fluid flow especially to avoid the secondary flow can be done. The knowledge will help in the modification and improvement of the pipe in terms of its diameter, pipe length and also the pipe configurations.

In addition, further works on this study and experiment can also be done. For example, an experiment to redesign the pipe in order to get and obtain a uniform fluid discharges from the nozzles. The data collected from this experiment could be used for designing a new pipe or nozzle that can lead for further modification. But it is necessary to have better experimental setup and some device of measurement to get better and accurate results.

REFERENCES

1. A. Aroussi, J. Roberts and P. Rogers (2004), *Investigation of secondary flows in a double pipe elbows: CFD study*, 11th International Symposium on Flow Visualization, Indiana, USA.

2. Aroussi, A. et al. (2004), *Investigation of secondary flows in a double pipe elbows: An experimental study*, 11th International Symposium on Flow Visualization, Indiana, USA.

3. F. Sánchez Silva, M. Toledo V. and J. Hernández Ruíz (1997), *Experimental study for the use of elbows as flowmeters*, 1997 ASME Fluids Engineering Division Summer Meeting.

4. Gerald L. Morrison (1997), *Flow field development downstream of two in plane elbows*, 1997 ASME Fluids Engineering Division Summer Meeting.

5. John A. Roberson and Clayton T. Crowe (1997), *Engineering Fluid Mechanics*, 6th Edition, John Wiley & Sons, Inc.

6. Joseph A. Schetz (1993), *Boundary Layer Analysis*, New Jersey, Prentice-Hall Inc.

7. Merle C. Potter and David C. Wiggert (1997), *Mechanics of Fluids*, 2nd Edition, Prentice-Hall, Inc.

8. Morris, P. et al. (1997), *The flow of a multiphase fluid through piping,* 1997 ASME Fluids Engineering Division Summer Meeting.

9. Rigit, A.R.H. et al. (2005), *Fluid flow simulation in a double L-bend pipe with small nozzle outlets*, 3rd International Conference on Computer Methods in Multiphase Flow, Portland, Maine, USA.

10. Ryuhei Yamaguchi, Takeshi Mashima and Yasunori Takahashi (1997), *Separated secondary flow and wall shear stress in side branch of right angle branch,* 1997 ASME Fluids Engineering Division Summer Meeting.

11. W. P. Graebel (2001), *Engineering Fluid Mechanics*, International Student Edition, Taylor & Francis Publishers.

12. William F. Hughes and John A. Brighton (1999), *Schaum's Outline of Theory and Problems of Fluid Dynamics*, 3rd Edition, McGraw-Hill Companies, Inc.

13. Yunus A. Çengel and John M. Cimbala (2006), *Fluid Mechanics; Fundamental and Applications*, McGraw-Hill International Edition, New York, The McGraw-Hill Companies, Inc.

APPENDIX A

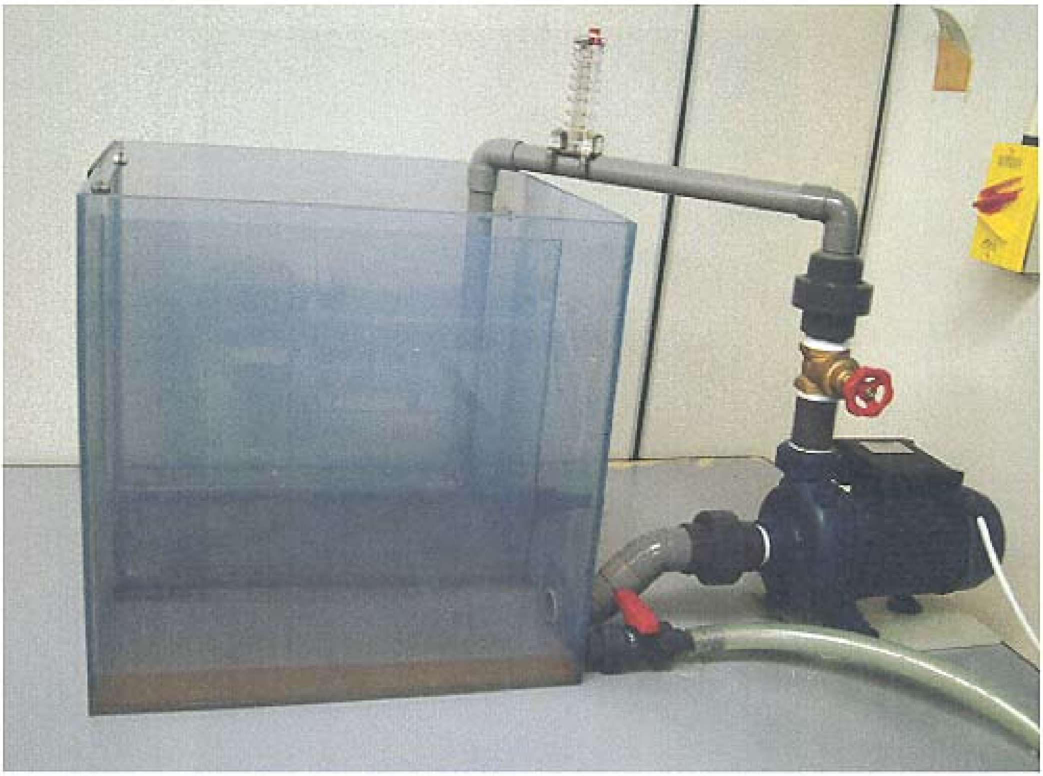

Figure A1: The actual experiment setup.

FigureA2: The pump for the experiment.

APPENDIX B

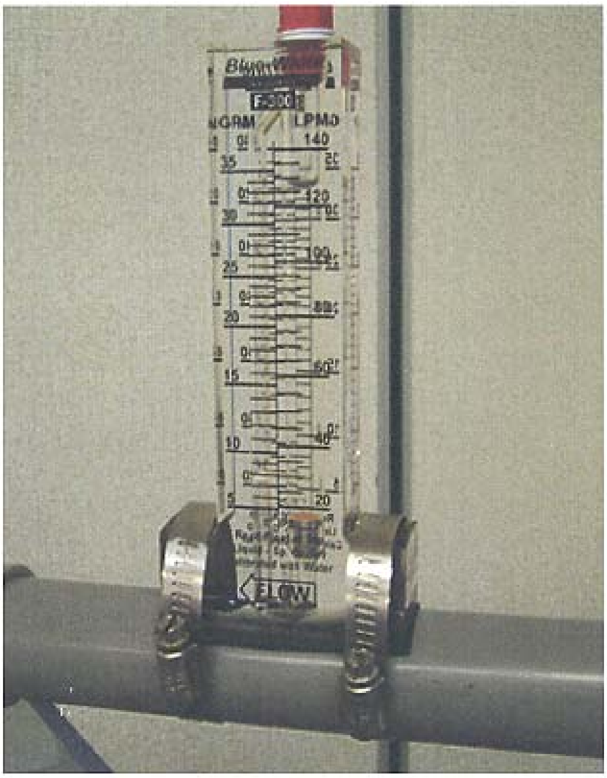

Figure B1: The actual flowmeter.

Figure B2: The control valve.